



Thermal Analysis for Diesel Engine Exhaust Manifold with Regard to the Boiling Phenomenon and Compared with Experimental Results

M. R. Assari*, S. Adeli, P. Nikandish

Department of Mechanical Engineering, Jondi Shapour University, Dezful, Iran

ABSTRACT: Exhaust manifold with the cooling system is widely used in diesel engines which use turbocharger. The appropriate solution to check how fluid passing through the manifold, is using the computational fluid dynamics laws. According to this, after 3D modeling of the manifold's body and make the appropriate meshing, using multiphase flows laws and Rensselaer Polytechnic Institute separation method for sub-cooled boiling at low pressure, the effects of fluid flow within the manifold have been simulated. In order to validate the method used for boiling, matching the results with the experimental results is performed. The information in this study includes the distribution of temperature, pressure, flow through the interior geometry and amount of vapor volume fraction that is created in the manifold. The results showed that in the inlet and outlet of manifold, there is a high temperature focus and consideration of boiling phenomenon instead of single-phase flow assumption, reduced the maximum temperature of these areas. The pressure which is applied to Manifold is caused by the combustion gases and the pressure of the cooling fluid is negligible in comparison with that. By analyzing the result, two critical areas of temperature focus were introduced and adaptation of cracks in the original sample with one of these areas, is indicating the accuracy and quality of the obtained results.

Review History:

Received: 6 May 2016
Revised: 16 August 2016
Accepted: 23 October 2016
Available Online: 9 November 2016

Keywords:

Exhaust manifold
Sub-cooled boiling
Computational fluid dynamic
Two-fluid boiling model
Rensselaer Polytechnic Institute separation method

1- Introduction

Exhaust manifold is attached at the top of the cylinder of the engine and its task is collecting the gases produced by combustion and transfers them to the turbocharger. Engine performance depends on the manifold to reduce thermal stresses by cooling block of engine and removal critical region in engine. Heat transfer between the cooling fluid and manifold is combination of convection heat transfer and nucleation boiling. When boiling develops beyond nucleate regime, the vapor film is formed on a solid surface and overheating occurs and may damage the equipment if boiling is not controlled. Modern engine cooling systems use sub-cooled boiling as an effective heat transfer form. The heat transfer rate achievable with boiling which is significantly higher, can keep the temperature in key thermal areas within an acceptable level. In this study, a three-dimensional two-fluid model coupled with Rensselaer Polytechnic Institute (RPI) wall boiling model was adopted to simulate the local two-phase characteristics of boiling under low-pressure condition. Sub-cooled boiling usually occurs when local boiling happens at the wall surface superheated by the high heat fluxes, even though the bulk flow has not reached the saturated temperature. Kurul and Podowski [1] proposed a new wall boiling model considering the bubble departure diameter, bubble detachment frequency, wall nucleation site density, bubble diameter, and other physical parameters with clear physical significance. Following studies such as Anglart and Nylund [2] implemented Podowski's two-fluid model into a CFD code to predict the distribution of void fraction in two-phase bubbly flows in fuel rod bundles. Yao and Morel [3] applied a two-fluid model to predict the

distribution of two-phase parameters in Freon R12 boiling flow. It has observed that the turbulent dispersion force plays an important role in determining the void fraction distribution and liquid temperature distribution. In RPI separation model for investigating the boiling phenomenon, liquid and vapor phases are solved separately by using two sets of governing equations. The interactions between phases are calculated based on the locally estimated bubble size and their amount [4]. In addition, many existing models are not suitable for low pressures because they have been developed for high-pressure state but working condition of manifold is in low pressure. In this situation we should use Unal [5] model to estimate the bubble departure diameter at low pressure working condition.

2- Mathematical Model

A multidimensional two-fluid Eulerian approach describes sub-cooled boiling flow in the cooling passages of exhaust manifold of the diesel engine. According to these assumptions, balance equations for mass, momentum and energy have been written separately for each phase. In addition, the liquid is treated as the continuous phase while the vapor bubbles are dispersed phase. Continuity equations for liquid and gaseous phase are as follows:

$$\frac{\partial(\rho_l \alpha_l)}{\partial t} + \nabla \cdot (\rho_l \alpha_l \bar{u}_l) = \Gamma_{lg} \quad (1)$$

$$\frac{\partial(\rho_g \alpha_g f_i)}{\partial t} + \nabla \cdot (\rho_g \alpha_g \bar{u}_g f_i) = S_i - f_i \Gamma_{lg} \quad (2)$$

Momentum equations for liquid and gaseous phase:

Corresponding author, E-mail: mr_assari@yahoo.com

$$\frac{\partial(\rho_l \alpha_l \bar{u}_l)}{\partial t} + \nabla \cdot (\rho_l \alpha_l \bar{u}_l \times \bar{u}_l) = (-\alpha_l \nabla p) + \nabla \cdot (\alpha_l \mu_l (\nabla u_l + (\nabla u_l^T))) + \Gamma_{lg} \bar{u}_g - \Gamma_{gl} \bar{u}_l + \bar{F}_{lg} \quad (3)$$

$$\frac{\partial(\rho_g \alpha_g \bar{u}_g)}{\partial t} + \nabla \cdot (\rho_g \alpha_g \bar{u}_g \times \bar{u}_g) = (-\alpha_g \nabla p) + \nabla \cdot (\alpha_g \mu_g (\nabla u_g + (\nabla u_g^T))) - \Gamma_{lg} \bar{u}_g + \Gamma_{gl} \bar{u}_l - \bar{F}_{lg} \quad (4)$$

Energy equation for liquid phase:

$$\frac{\partial(\rho_l \alpha_l h_l)}{\partial t} + \nabla \cdot (\rho_l \alpha_l \bar{u}_l h_l - \lambda_l \alpha_l \nabla T_l) = \Gamma_{lg} h_g - \Gamma_{gl} h_l + Q_l \quad (5)$$

According to the mechanical model on sub-cooled boiling flow of Kurul and Podowski [1], the wall heat flux (Q_w) is divided into three different components: the single-phase convection heat flux ($q_{l\phi}$), the quenching heat flux (q_Q) and the wall evaporation heat flux (q_e), as shown in the Fig 1.

$$q_w = q_{l\phi} + q_e + q_Q = S_l \rho_l c_{pl} A_{l\phi} u_l (T_w - T_l) + 2f \left(\frac{\tau_Q \lambda_l \rho_l c_{pl}}{\pi} \right)^{0.5} A_{bub} (T_w - T_l) + \frac{N_a f \pi d_{bw}^3 \rho_g H_{lg}}{6} \quad (6)$$

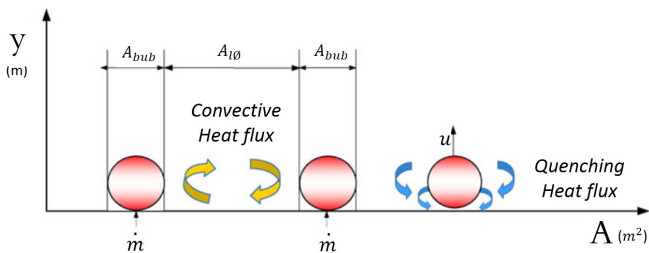


Figure 1. Heat flux partition of wall boiling model.

Unal [5] assumed that a bubble is subjected to the simultaneous micro layer evaporation at the bottom and to condensation at the top. The bubble may slide or collapse on the heated surface during its lifetime. After reaching the maximum size, the bubble departs from the wall and migrates into the sub-cooled liquid, where the bubble can collapse. Pressure, material of solid wall, wall superheating, liquid sub-cooling and local velocity influence the bubble size at departure.

3- Results and Discussions

By simulating the fluid flow in manifold, many information about it was obtained. This information includes the distribution of temperature, pressure, flow through the interior

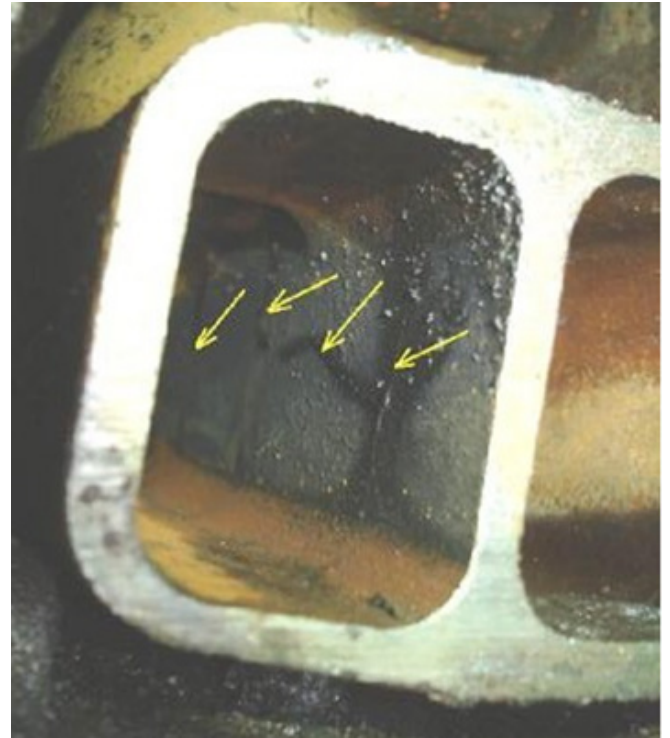


Figure 2. Crack in experimental model

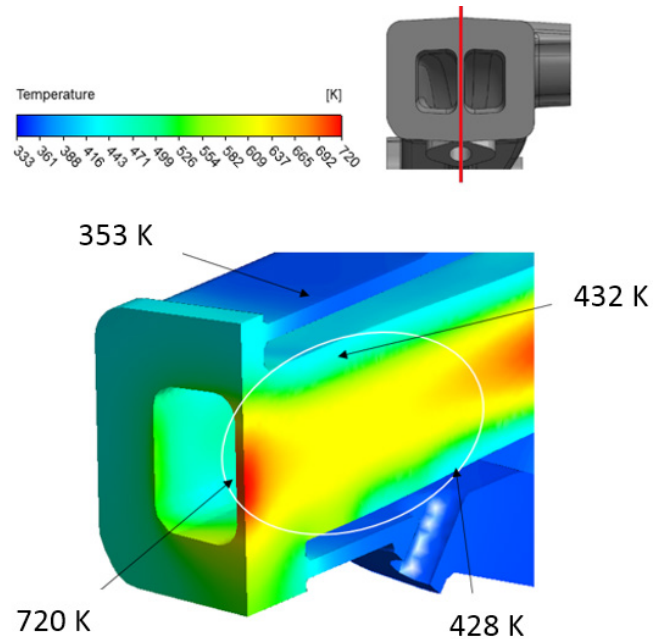


Figure 3. Temperature distribution in manifold

geometry and amount of vapor volume fraction that created in the manifold. The results showed that in the inlet and outlet of manifold, there is high temperature focus and consideration the assumption of boiling than single-phase flow, reducing the maximum temperature of these areas. Industrial units that had used this type of manifold, report outlet water temperature 80°C and the results of the simulation shows this value around 85°C. This slight temperature differences can occur for the following reasons:

1. Ignore the sediment existence in the inner surfaces of the

manifold.

2. Using the standard properties of cast iron in the body of the manifold in simulation and possibly the difference between them and properties in the original model
3. Errors in temperature measurement equipment fluid and user error in data record.

By analyzing the results, two critical areas of Temperature focus were introduced and adaptation of cracks in the original sample with one of these areas, indicates the accuracy and quality of the obtained results.

4- Conclusions

Using Computational Fluid Dynamics is reliable solution to find critical conditions in industrial applications. Using this technique allows the designers to develop this industrial equipment with high working efficiency, without spending exorbitant costs of making and testing different samples.

References

[1]Kurul, N. and Podowski, M. Z., On the modeling

of multidimensional effects in boiling channels, ANS Proc.27th *National Heat Transfer Conference*, Minneapolis, MN, July 28-31, 1991.

- [2] Anglart,H, Nylund.O. CFD application to prediction of void distribution in two-phase bubbly flows in rod bundles. *Nuclear Engineering and Design*, 1996,163(1–2):81-98
- [3] Yao W, Morel C. Prediction of Parameters Distribution of Upward Boiling Two-Phase Flow With Two-Fluid Models. *International Conference on Nuclear Engineering*. 2002:801-808.
- [4] Puneekar, H, Das.S, Numerical Simulation of Sub-cooled Nucleate Boiling in Cooling Jacket of IC Engine. *SAE paper*: 2013-01-1651; 2013.
- [5] Unal,H, Maximum bubble diameter, maximum bubble growth time and bubble-growth rate, *Int. J. Heat Mass Transfer* 643–649,1976.

Please cite this article using:

M. R. Assari, S. Adeli, P. Nikandish, Thermal Analysis for Diesel Engine Exhaust Manifold with Regard to the Boiling Phenomenon and Compared with Experimental Results, *Amirkabir J. Mech. Eng.*, 50(4) (2018) 711-726.
DOI: 10.22060/mej.2016.786



